AUTODESK Instructables

Make a 3D Print From a 2D Drawing

By safay in Workshop3D Printing

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Introduction: Make a 3D Print From a 2D Drawing

Using a few open source software tools and very little technical knowledge, it's possible to turn a 2D vector drawing into an extruded object for 3D printing. I used this technique to make a bracelet.

Overview:

1. Make an .svg file of the shape you want to extrude
2. Convert it to OpenSCAD format using an Inkscapeextension
3. Use OpenSCAD to render and generate an .stl file
4. Print the object

Step 1: Make an .svg File of the Shape You Want to Extrude

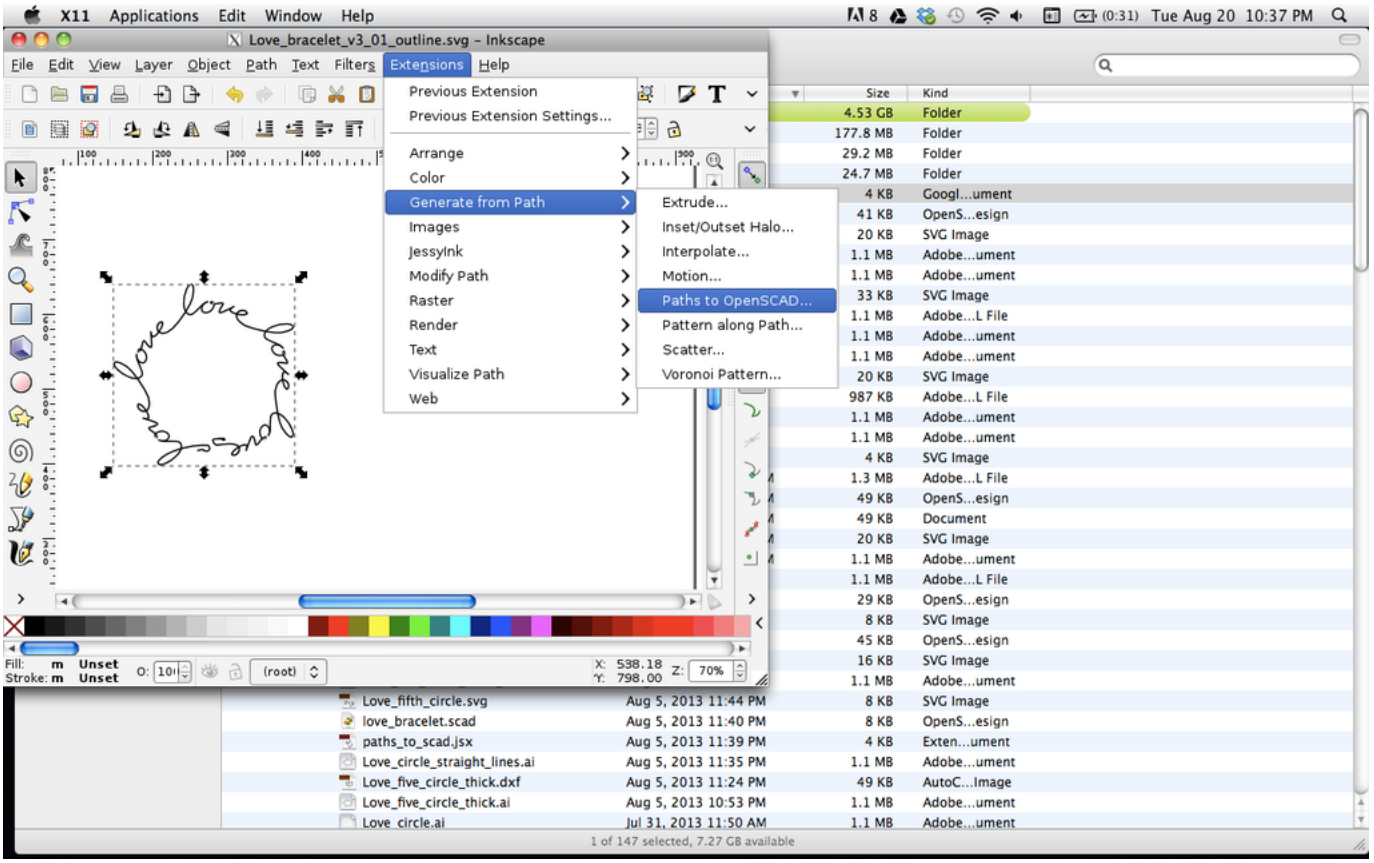
First get a shape that you want to extrude. It needs to be in a vector graphics format, not bitmap (or raster) format. Read up if you're not familiar with these terms.

To get a vector graphic you have three options:

1. **Download it.** You could use Google Advanced Image Search and limit your search to .svg files, a common vector graphics format on the web. You want to make sure you have permission before you use anyone else's drawings, though.
2. **Draw it yourself.** Use Inkscape, Illustrator, CorelDRAW, or any of the many other vector graphics applications. Drawing vector graphics is its own topic worthy of many instructables.
3. **Make a digital trace.** If you do have a raster image (bitmap), then you can convert it to a vector graphic with Illustrator's live trace or Inkscape's trace features, which are fun to play with. Keep in mind that these features are not magic, and you'll have to fiddle around a lot to get the graphic right.

Then save the file as .svg format.

Step 2: Convert the .svg to OpenSCAD Format



OpenSCAD is the software that will extrude the object for us. However, we first need to convert the .svg to OpenSCAD format in Inkscape using the magic of this Thing.

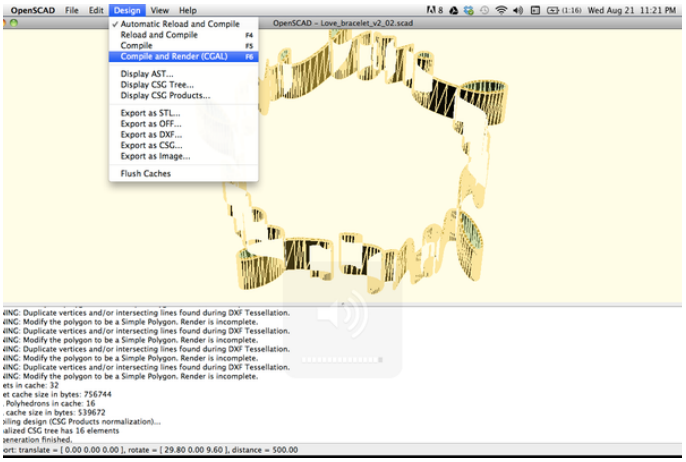
Do this:

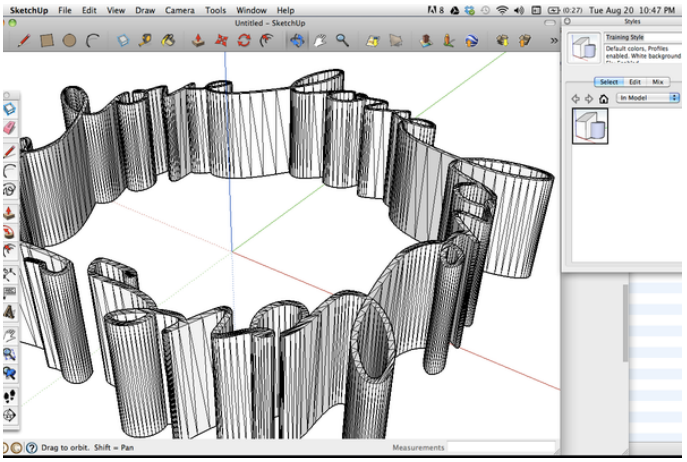
Install Inkscape.
Download the paths2openscad script at GitHub.
Move the script to your ~/.config/inkscape/extensions folder.
Open Inkscape.

File -> Open to open the .svg file.
Extensions -> Generate from Path -> Paths to OpenSCAD to generate the .scad file.
Save the file with the extension ".scad"

Now you're ready to render.

Step 3: Render and Generate an .stl File





Now download and install OpenSCAD. OpenSCAD is a CAD program that doesn't have an interactive graphical user interface for drawing objects. Instead it uses its own scripting language. Fortunately for our purposes here you don't need to learn the language. But now that you know it exists, you're probably going to want to learn it. Right?

Getting down to business in OpenSCAD:

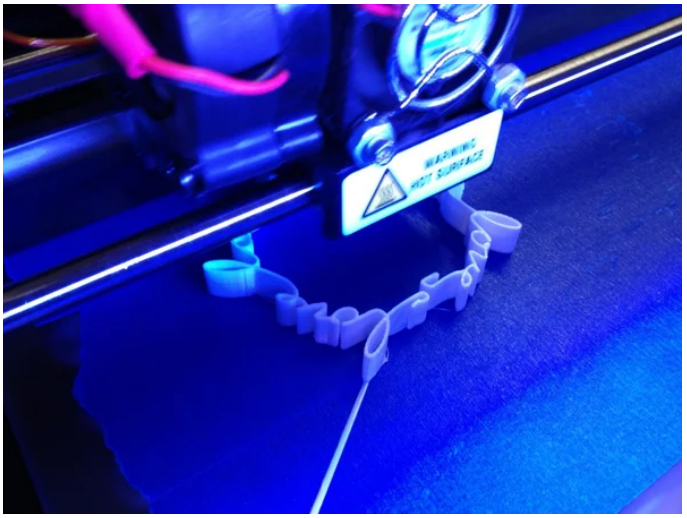
File -> Open to open the .scad file made in the last step. If you get warnings, just ignore them.

Design -> Compile and Render to build the object. This might take a while, depending on how complex your object is.

Once the rendering is complete, ***Design -> Export to STL*** to export the file for printing.

The .stl file should be ready to print! However, if you first want a better look at your rendered object, you could open it up in SketchUp.

Step 4: Print Your Object



Print your object. Here you can see my bracelet being printed on the MakerBot Replicator 2, which Instructables hooked us up with here at Ace Monster Toys.

If you don't have a 3D printer, have Shapeways print it for you, build a printer, buy one, or better yet, join your local hacker/makerspace or TechShop and use theirs.

Thank you:

davr for showing me how to use a 3D printer
dnewman for developing and sharing the OpenSCAD converter
heilis for playtesting the bracelet
Ace Monster Toys for being an awesome place